(511)

199101078Y

TURBOMACHINERY

Anthony J. Strazisar, Raymond E. Gaugler,
John J. Adamczyk, and Dale J. Arpasi
NASA Lewis Research Center
Cleveland, Ohio

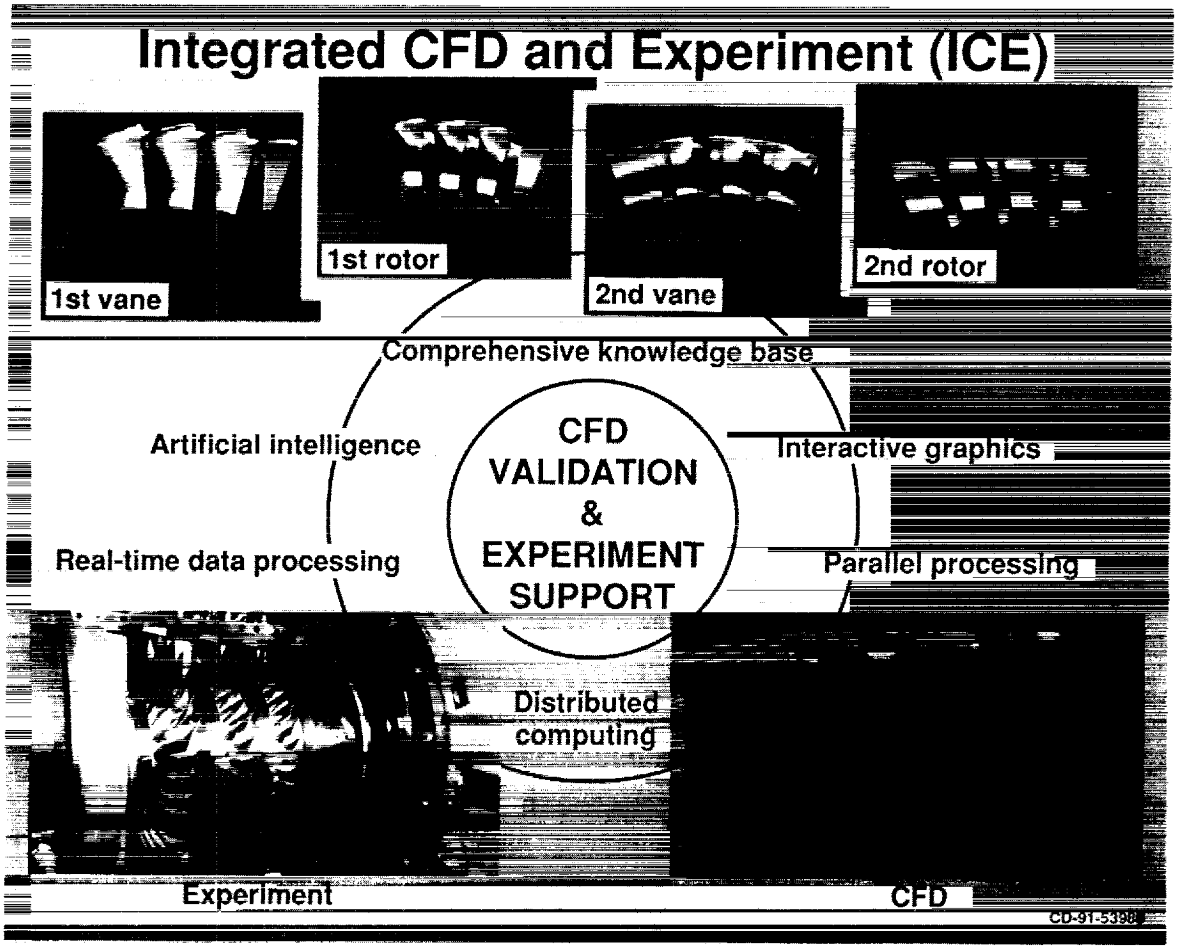
and

Isaac Lopez
Propulsion Directorate
U.S. Army Aviation Systems Command
Lewis Research Center
Cleveland, Ohio

We are responsible for developing the computational capabilities and furthering the understanding of flow physics phenomena fundamental to the advancement of turbomachinery technology. Our research involves the development, assessment, and application of computational fluid dynamics (CFD) tools for turbomachinery flows and the acquisition and analysis of experimental measurements of flows in turbomachinery. The measurements involve both simplified and realistic, complex geometries, and are used both for the validation of advanced numerical flow analysis codes and for the development of physical models for flow phenomena. They are obtained in both large scale, low speed facilities and full scale, high speed, transonic facilities. The computational emphasis involves the development of advanced computer codes, their assessment by comparison with quality data, modification of codes to extend range and accuracy, and application of codes to practical problems to demonstrate their value in design and in providing detailed description of flow field phenomena. The capability of the CFD codes to accurately calculate losses and heat transfer in 3D viscous steady and unsteady flows is emphasized. Through interactions with other Lewis organizations, these capabilities are applied toward the solution of technical problems important to the mission of the NASA.

Our research involves experiments using advanced instrumentation systems such as laser and hot wire anemometry in order to resolve spatial and temporal flow field variations. Major facilities include a full scale axial flow compressor, a large, low speed centrifugal compressor, a low speed multistage axial research compressor, and a full annular turbine stator cascade.

We synergistically apply experimental and computational efforts to maximize our understanding of the complex flows existing in turbomachinery passages. We apply 3D viscous steady state computational codes to both axial and radial turbomachinery with emphasis on the prediction of the detailed blade row performance and local heat transfer coefficients. We apply 2D and 3D unsteady codes to evaluate the impact of flow field deterministic unsteadiness on stage performance.



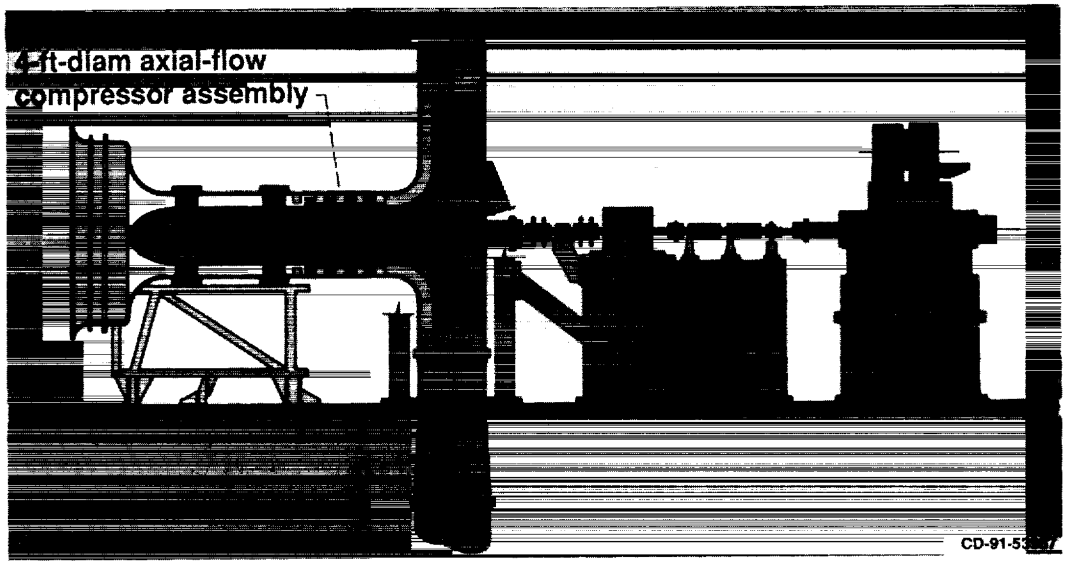
The ICE Program was created to meet the needs of research programs such as the Multistage Compressor Flow Physics Program which feature tightly coupled experimental and computational research efforts. ICE brings computer science disciplines such as parallel processing, interactive graphics, and artificial intelligence together with concepts such as distributed computing and knowledge bases to create a unique hardware/software capability.

The ICE program has several key elements. The first element involves the development of methods for parallelizing CFD codes and executing them on parallel processors, which are expected to offer supercomputer performance at departmental computing costs in the near future. The second element involves using parallel processing techniques coupled with workstation graphics to rapidly reduce and display measurements from advanced instrumentation systems such as laser and hot wire anemometers. This capability will allow experimental researchers to analyze complex data sets in near real-time, while the test rig is still on-condition, and make adjustments in measurement locations or extent in order to capture flow physics details. The third element of the ICE program involves storing CFD results and experimental data as independent data bases on a common mass storage system, then retrieving both CFD and experimental results and displaying them through a common graphics interface for further off-line comparison and analysis.

Multistage Compressor Flow Physics

Low Speed Multistage Axial-Flow Compressor

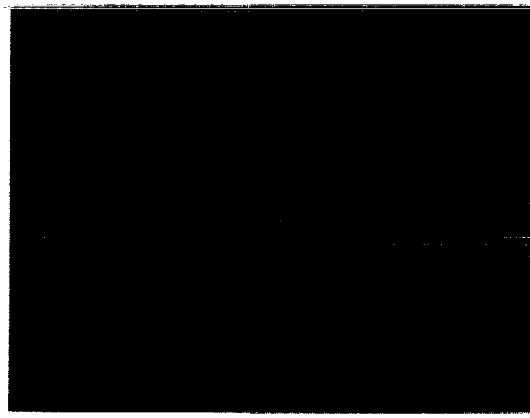
Detailed mechanical design completed	1989
Fabrication and assembly	1990-1991
Operational	1992



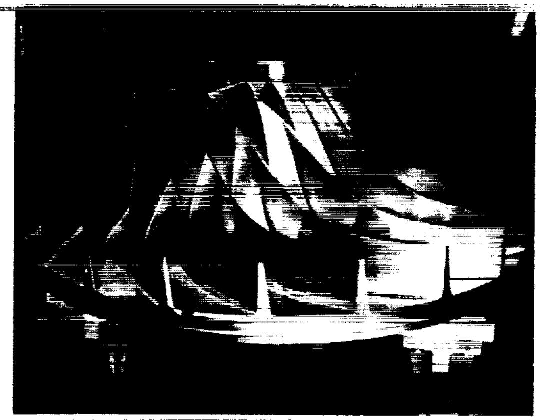
NASA Lewis has in place a Multistage Compressor Flow Physics Program which is aimed at improving our understanding of the complex mixing phenomena which control the performance of multistage compressors. The program features a tightly coupled effort between computational, experimental, instrumentation, and high-speed-computing research areas.

The experimental research will be conducted in both high-speed and low-speed compressors. The high-speed research, focussed on the measurement of thermodynamic properties through the compressor, will be conducted in existing Lewis facilities. The low-speed research will focus on a detailed study of mixing phenomena and will be performed in a new low-speed multistage axial compressor. A four-stage axial-flow compressor research package is currently being fabricated and will be interchanged with a large centrifugal compressor impeller which is currently installed in the low speed compressor facility. The axial research package is 4 ft in diameter and rotates at 950 rpm. The large size, low speed, and long inlet result in thick end wall boundary layers, the proper blade chord Reynolds number, and minimal blockage effects associated with intrusive instrumentation. The research package is designed to enable changes in blade stagger, solidity, and relative circumferential positioning between blade rows, features both casing-mounted and rotor-mounted aerodynamic survey capability, and also features optical access for non-intrusive measurement techniques and flow visualization techniques.

Low Speed Centrifugal Compressor Research



3D Navier-Stokes code predictions of fluid particle paths



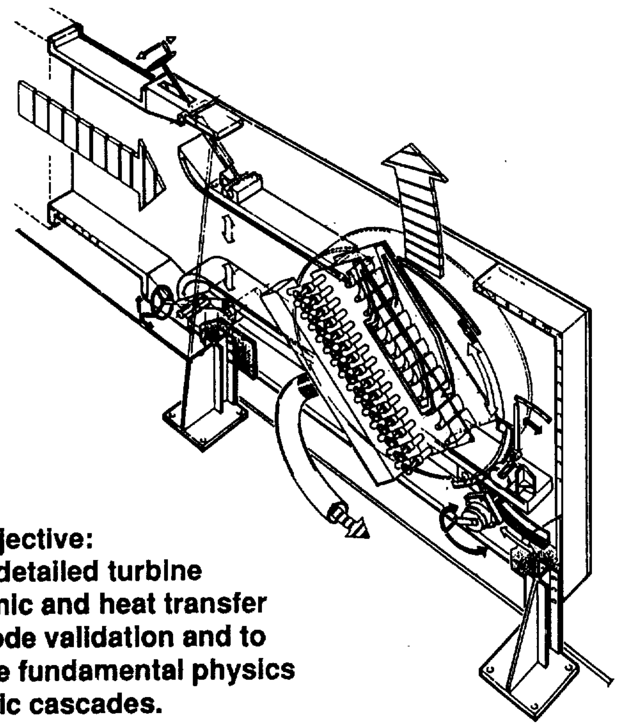
5-ft-diam impeller

CD-91-53988

Numerical analysis of centrifugal compressor flow fields using 3D Navier-Stokes codes is being coupled with experimental measurements acquired in a unique large-size, low-speed centrifugal compressor impeller in order to improve our understanding of centrifugal compressor flow physics and to assess the accuracy of 3D viscous codes. The low-speed impeller has a tip diameter of 60 in. and a rotational speed of 1950 rpm. The large size and low speed of the impeller yield tip clearance flow features and viscous flow regions such as blade and end wall boundary layers which are large enough to measure with laser anemometry.

A photograph of the low-speed impeller is shown above, along with the results of a numerical prediction of the impeller flow field, including tip clearance effects, generated with a 3D Navier-Stokes code. The illustration above is aimed at visualizing the secondary flow field within the impeller. Fluid particles in the blade boundary layer are being tagged from hub to tip near the blade leading edge on both the blade pressure and suction surfaces and then followed as they proceed toward the impeller exit. The numerical result indicates that this fluid is being driven to the tip of the blade, then passes through the tip clearance gap, and accumulates in a vortex-like structure which exits the impeller near the pressure-surface side of blade passage. Such results improve our understanding of the complex impeller flow physics and serve as a guide in planning the location and extent of detailed laser anemometer flow field surveys.

Linear Transonic Turbine Cascade



Project objective:
To obtain detailed turbine aerodynamic and heat transfer data for code validation and to explore the fundamental physics of transonic cascades.

CD-91-53989

To cut engine weight and cost, the trend in future turbine design will be toward fewer turbine stages. This means more work per stage, which means higher turning angles and transonic exit flows. Design rules based on empirical models predict that these types of blades will suffer high losses while CFD codes do not. Ever increasing turbine inlet temperatures require the accurate heat transfer predictions be made.

The test program had two objectives. First, appropriate loss models for advanced blade designs at high Mach numbers would be determined. Second, and more importantly, a database of experimental data would be obtained for code validation. Codes such as RVC3D and PROTEUS, which are designed to accurately predict the aerodynamics for transonic flow fields, would benefit from a transonic field database.

Automated Inverse Airfoil Design Code

- **Rapid generation of 2D airfoil shapes with prescribed aerodynamic characteristics.**
- **Provides baseline design configuration beyond existing data base.**

CD-91-53990

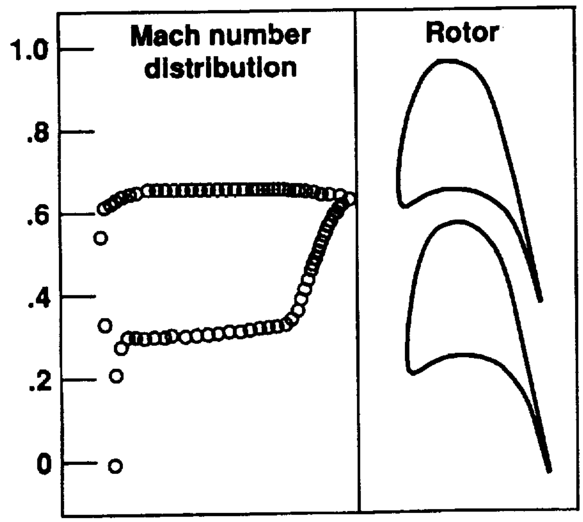
An original, innovative approach to the design of airfoils has been developed. Airfoil sections, whether for aircraft wings, turbomachinery, or numerous other applications, have historically been designed by a trial-and-error process. That is, the engineer or designer successively guesses different airfoil shapes and then, by analysis or experimentation, determines if the required performance is obtained. This performance is often measured in terms of the flow turning achieved by the airfoil and by the pressure distribution on the airfoil surface, which can be directly related to the lift and drag generated. This process is labor intensive due to the manual trial-and-error process employed.

Here at NASA Lewis, Dr. Jose Sanz has developed an automated approach that generates the airfoil shape that will satisfy the required performance as specified by flow turning and airfoil pressure distribution. This automated procedure builds on the Inverse Hodograph Method.

To further explore the range of application of the code after the incorporation of the automated procedure, which might be of interest to the engine industry in the specific field of axial turbomachinery, a set of cases has been supplied by different companies to be tried at NASA Lewis with the automated code. The information supplied consists mainly of the corresponding velocity triangles for each test case. These turbomachinery test cases include both axial compressor and turbine blading.

Automated Inverse Airfoil Design Code

- **Used to generate baseline design for an advanced turbopump turbine**
- **Design outside industry database (160° turn)**



Design parameters

Inlet Mach No.	0.548
Exit Mach No.	0.643
Inlet Flow Angle	80.11
Exit Flow Angle	–80.08
Solidity	1.372

CD-91-53991

A good demonstration of the value of the inverse design code occurred recently. The NASA Marshall Spaceflight Center had contracted with industry for the design of a generic high-pressure fuel turbopump turbine. The parameters specified for the turbine resulted in design requirements that were outside the range of industry experience. The contractor was unable to develop a baseline shape that met the requirements. Subsequently, the contractor spent two days here at Lewis working with Dr. Jose Sanz to generate the required baseline design. Since then, the contractor has been able to complete the design and release it to the computational fluid dynamics community for analysis.

CONCLUDING REMARKS

In summary, over the last few years we have seen a dramatic rise in the capability to compute the 3D flow in turbomachinery, and we have developed a program where we synergistically apply experimental and computational efforts to maximize our understanding of the complex flows existing in turbomachinery passages. This improved capability and understanding will lead to significantly better turbomachine designs in the future.